

WORKSHOP FOR CFD APPLICATIONS IN ROCKET
PROPULSION AT NASA-MSFCMehtab M. Pervaiz
April 20, 1993

516-34

13-13-0

P-20

Title: An Inducer CFD Solution and Effects Associated With Cavitation

This presentation describes a CFD analysis for an Alternate Turbopump Development (ATD) configuration. The analysis consists of a coupled configuration of the inducer and impeller. The work presented here is a joint collaboration of J. Garrett, J. Kuryla and myself.

Outline:

This view graph provides an outline of the current presentation. I will start with the ATD configuration for the inducer and impeller and the corresponding CFD solution. Subsequently, I will describe the current cavitation modeling approach that has been utilized on this configuration. A review of various cavitation modeling approaches will then be presented. Various suggestions and modeling ideas will then be presented for analyzing cavitation. The talk concludes with a brief summary and future plans.

14.6 Degree Inducer-Impeller with Splitters

This view graph shows the computational mesh for the inducer-impeller combination of the hub surface. The inducer and impeller rotate at the same RPM, with no clocking in between them. The configuration consists of a full inducer and impeller coupling with a continuous main blade that proceeds from the inducer leading edge to impeller trailing edge. A single set of splitters is apparent in the impeller. The figure shows the blades and splitters projecting out of the hub surface. Although there are four total main blade passages, only one was considered for analysis and periodic boundary conditions were applied before the leading edge of the inducer and after the trailing edge of the impeller main blade. There are 18 computational nodes in between the main blade passages. There are about 110 points along the flow direction.

Summary of Pressure Distribution Over the Full Configuration

This shows the pressure distribution on the full configuration (computational results produced with a multiplicity of four) with splitters. The top two figures show the pressure distribution on the suction and pressure sides of the blades and splitters and the hub surface is shown shaded. The bottom figures shows the pressure distributions on the hub, centerline and shroud surfaces whereas the blades and splitters shown in the grey shade. These figures indicate the existence of negative pressure near tip of the suction side of the inducer blade. The negative pressures are due to the fact that the CFD model assumes a single-phase flow and is incapable of modeling vapor where the local static pressure becomes lower than the vapor pressure.

Current Cavitation Model Applied to ATD Configuration

This viewgraph presents an approach that we have utilized at Pratt & Whitney for getting an estimate for the change in loading attributed to cavitation. The first step is to carry out a single phase 3-D CFD solution for the configuration at hand. For the ATD configuration, as shown in the previous figure, the CFD analysis shows that the static pressure on the suction side near the tip region becomes lower than local vapor pressure. Hence the effect of cavitation must only manifest itself near the suction side tip region and gradually diminish as one move towards the hub. The next step is to compute a 2-D potential flow cavitation sheet model corresponding to the conditions relevant at the blade tip. This potential flow cavitation sheet model has been developed by Penn State University for P&W. The model yields *cavitation correction factors*, Δp_{sheet} , for *pressure loads* as a function of tip chord length. These corrections must be multiplied with non-cavitated blade delta-pressure to obtain cavitated delta-pressure. Note that p is used here for pressure corrections, whereas P is used for static pressures. Next the tip sheet cavitation correction factors are scaled linearly such that no cavitation correction is needed at the hub. This yields a 2-D distribution of the correction factors, Δp_{cor} , on the blade surface. Thus in

$$\Delta p_{cor} = K(s, r) \Delta p_{sheet}$$

$K(s, r)$ represents the linear scaling transformation as a function of local arc length s measured from the inducer LE for a streamline at a local radius r and Δp_{cor} represents the multiplier for delta-pressure at all blade locations. Once the functional form of the delta-pressure corrections is known, it is multiplied at all the CFD blade locations

to obtain a new distribution ΔP_{cor} for blade loads. This distribution is then used to compute the new suction side pressure corresponding to the CFD solution. The suction side pressure is given by:

$$P_s = P_p - \Delta P_{cor}$$

where P_p represents the pressure on the pressure side of the blade. As a sanity check, one should verify that the corrected pressure on the suction side remains above the vapor pressure. If this is not achieved, then the 2-D sheet model should be repeated for other streamlines and the cavitation corrections determined by a bilinear interpolation.

Distribution of Cavitation Correction Factors

This viewgraph shows the functional form of the cavitation correction on the blade tip as a function of local arc length along the blade. The effect of factors greater than unity is to increase the pressure load downstream of the bubble, whereas for those locations where the bubble actually exists, the effect is to decrease the pressure loads. This can be thought of as a blockage effect which causes the variations in pressure. The figure also shows the effect of linearly scaling the correction factors and its diminishing effect as one goes from tip to hub. Note that this procedure should be regarded as rough estimate of the changes associated with cavitation. For this reason a more elaborate non-linear transfer function is not warranted. Ideally the cavitation correction procedure should be a part of the CFD numerics.

Inducer Blade Delta Pressures

This figure shows a projected view of the inducer blade in the x-r plane. The top figure shows the blade pressure loads as predicted by the CFD code and before the cavitation correction is applied. The cavitation bubble is limited to the red region near the blade tip. The lower figure shows the pressure loads after the cavitation correction is applied. The effect of this correction is to locally decrease the load due to the blockage and to increase the loads downstream of it. The saw-tooth behavior relates to the axial coarseness of the mesh when the tip correction factors are linearly transferred to other locations.

Inducer Blade Suction-Side Pressures The top figure shows the blade suction

side pressures as predicted by the CFD code and before the cavitation correction is applied on a projected x-r plane. Note that the pressures are substantially negative in the cavity region. The lower figure shows the suction side static pressure after the cavitation correction is applied. In this case all the pressures have become positive. The blockage effect is felt to about mid-span location at the tip. The blockage effect diminishes as one moves from tip to hub.

Linear Cavitation Model

I will now present some sheet cavitation models that one can consider. Some of these have been considered by Penn State. Additional ideas are presented that can provide a more accurate and simpler treatment of the cavitation problem. This viewgraph describes the linear cavitation sheet model. In such a model the cavity interface is treated as a streamline with a constant pressure equal to the vapor pressure. The approach is analogous to classical linear theory used in the potential flow codes. Interface conditions are transferred to the solid body and zero normal velocity condition is relaxed at the solid boundary. Thus the pressure of the solid boundary adjacent to the bubble will be equal to the vapor pressure. The relaxing of zero normal velocity condition at the solid boundary simply discards all flow quantities interior to the cavity and correct blockage effects are simulated outside of the cavity. The disadvantage of the approach is that it does not explicitly treat the vapor phase and that the approach is not robust. Since the approach handles only the liquid phase, the modeling can not be accurate for cryogenic fluids operating near the critical point. There can be certain cavity termination problems in 2-D which can only become worse in three spatial dimensions.

Non-Linear Cavitation Model

In the non-linear cavitation model, the cavity interface is again regarded as a streamline with a specified cavitation pressure. The basic difference between the linear and non-linear models is that the computational domain evolves with the solution procedure as a sequence of linear solutions followed by modifications of the domain to accommodate the cavity geometry. Thus the computational domain is regridded after locating the positions where the cross-over below the vapor pressure occurs. These cross-over points are recomputed subsequently for a better definition of the cavity. One can, in principle, iterate to a "correct" cavity geometry via these multiple passes. The disadvantage of the

approach is that mesh movement may introduce significant grid distortions and the grid lines may actually cross for complicated configurations. Significant man power efforts are already spend in gridding complicated configurations, it will make sense to retain these meshes for a cavitating flow. Hence a non-linear cavitation approach that holds the base grid will be highly desirable. It should also be pointed out that regridding may only be suitable for box-like domains and may have significant problems for complex 3-D configurations. For example, there can be multiple cavities in a domain, or a cavity may be completely embedded in the domain and not hooked to a physical boundary. There are other disadvantages with respect to the physics of cavitation sheet models. If $p > p_v$, then the fluid remains a liquid; otherwise the fluid is in the vapor phase. No dynamics of the vapor phase are included in the computations, so for example, the change in volume in flashing to a vapor, or the collapse in volume due to condensation is not taken into account. These effects can be accounted by considering a multi-phase approach with more elaborate description of physics. However, the incorporation of two-phase flows can only be accommodated in a regridding methodology with some difficulty.

Proposed Cavitation Model

I will now present a non-linear two-phase sheet model in which a structured grid does not have to be regenerated. Firstly, for the sake of all subsequent discussion, consider the static pressure to be referenced from vapor pressure. Thus pressures below the vapor pressure are regarded as negative pressure for the liquid. First carry out a single phase (liquid) CFD solution to convergence. Regard cells with positive pressure on all nodes as liquid cells, regard cells with all negative node pressures as vapor cells for subsequent iterations, and regard all remaining cells as interface cells where both liquid and vapor fluxes will be carried out simultaneously. The third step is to determine the liquid vapor interface as zero crossing for pressure on the edges of interface cells. Consider a 2-D cell $acef$ shown in the viewgraph for reference. Treat the liquid-vapor interface bd locally as a streamline. This means that one should discard the velocity components normal to the streamline. Suppose u_b^p is the predicted value of velocity at the zero crossing point b based upon linear interpolation of the edge values ac . Let u_b^c be the corrected value of the velocity that discards the velocity component normal to the line bd . Then redo the flux balance for the liquid part of the cell on nodes $abdef$. Except for the pressure terms in the momentum and energy equations, the interface bd yields a zero flux. Thus

the flux balance for x-momentum equation on the interface bd is simply:

$$\int_{bd} (\rho u^2 + p) dy - \rho u v dx = p_v \Delta y_{bd}.$$

Based upon the new flux balance, one can compute the “distribution formulas” for nodes aef . These distribution relations yield contribution of the cell flux balance to the corresponding nodes (See Ni’s classical AIAA Journal Paper, 1985). A similar flux balance can be carried out for the vapor part of the cell. One can, in fact, regard the interface cell nodes to be multiply defined for vapor and liquid parts. Thus the liquid part will have real fluid values at nodes aef and fictitious values at node c . The flux balance can be carried out independently based upon the real and fictitious values and the nodes updated separately. This approach constitutes a simple two-phase approach to cavitation without the mesh regridding option. Note that I have not specifically stated anything about the physics of two-phase flows yet. A more elaborate two-phase approach without appropriate physics may only yield qualitatively correct answers. This follows on the next viewgraph.

Additional Modeling Enhancements

A more elaborate equation of state which is approximately valid in the gas/liquid two-phase region, as well as at moderately high temperatures is needed. A suitable model that describes the law of corresponding states can be easily implemented for the thermal equation of state (that relates pressure, density and temperature). This has to be coupled with a more representative caloric equation of state (that relates specific heat to temperature). One has to also employ a more accurate energy equation to describe two-phase flow physics. This basically includes latent heat of vaporization for the vapor phase. Certain cryogenic fluids, with critical temperatures below 50 K, exhibit quantum effects, and do not necessarily satisfy the principle of corresponding states. A more elaborate equation of state, possibly in the form of look up tables, may be needed for these fluids. Note that both liquid hydrogen and helium exhibit these quantum effects. Liquid hydrogen is a common fluid for rocket engine fuel pumps. Lastly, one can utilize mesh adaptation to locally subdivide the cells in the vicinity of liquid-vapor interface to impart additional spatial resolution. This will eliminate the need to carry out multi-valued flux balance, providing that the embedded cells are reasonably resolved. Note that embedded mesh adaptation does not regrid the “base” mesh, the skewness of the embedded meshes will only be limited by the skewness of the coarse mesh, and that cells can never cross if the base mesh is not crossed over. Pratt & Whitney already has

suitable data base structure for handling this approach.

Closure

A CFD solution for ATD inducer/impeller pump configuration, with splitters, has been presented at the design point. This solution points to the existence of a cavitation bubble in the domain. Current cavitation approach employs a separate potential flow cavitation sheet model that yields cavitation correction factors for the CFD predicted pressure loads. Penn State University is currently studying linear and non-linear approaches for handling 3-D configurations. If a suitable linear cavitation approach can constitute a simple and effective model, we will implement such a model in the Pratt & Whitney design code. Additional enhancements for two-phase flows and more accurate physics will be considered as part of a long-term effort.

AN INDUCER CFD SOLUTION AND EFFECTS ASSOCIATED WITH CAVITATION



**M. Pervaiz
J. Garrett
J. Kuryla**

**Pratt & Whitney, GESP
West Palm Beach, Florida**

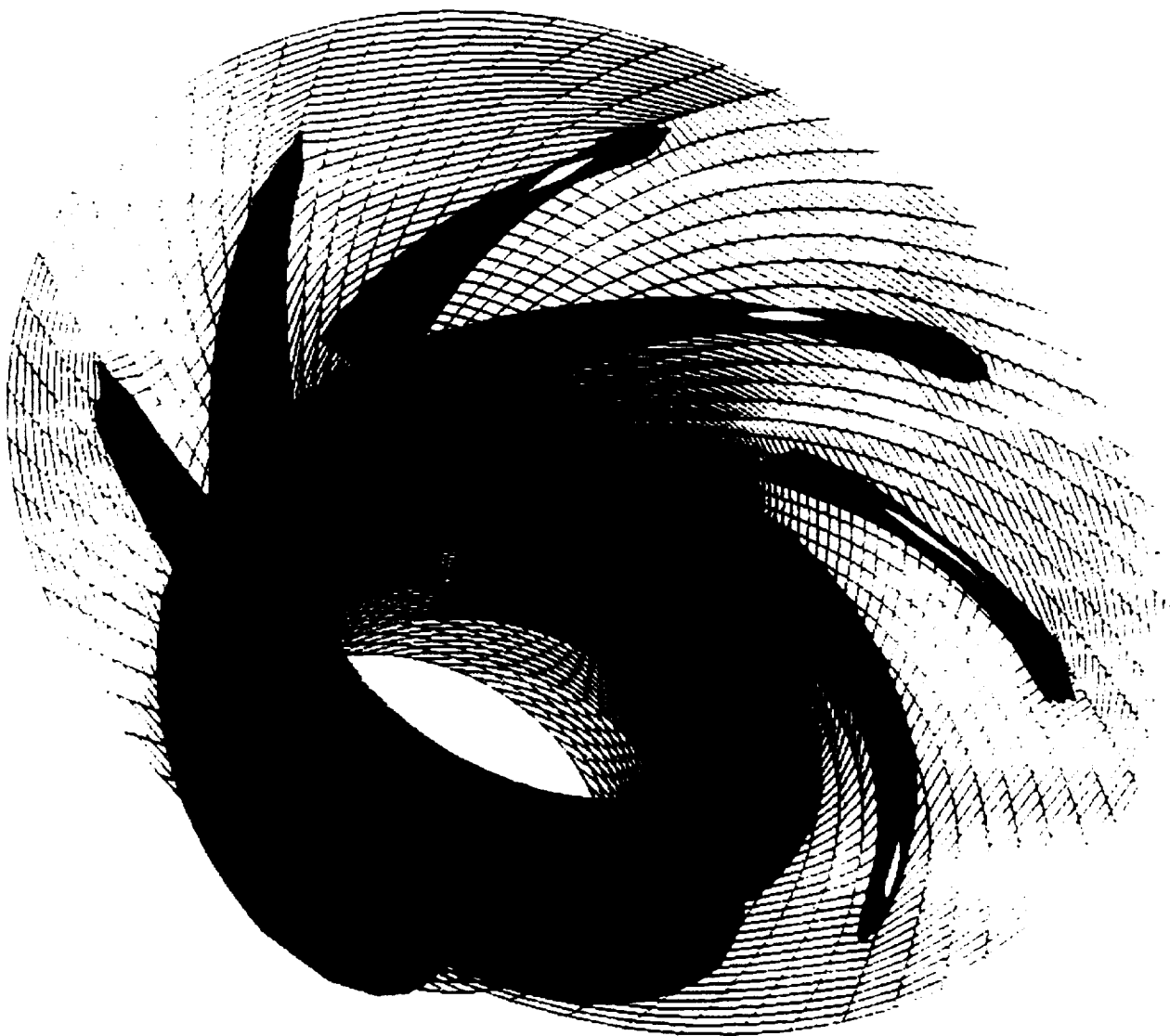
April 20, 1993

**Presented At Workshop for CFD
Applications in Rocket Propulsion, NASA-MSFC**

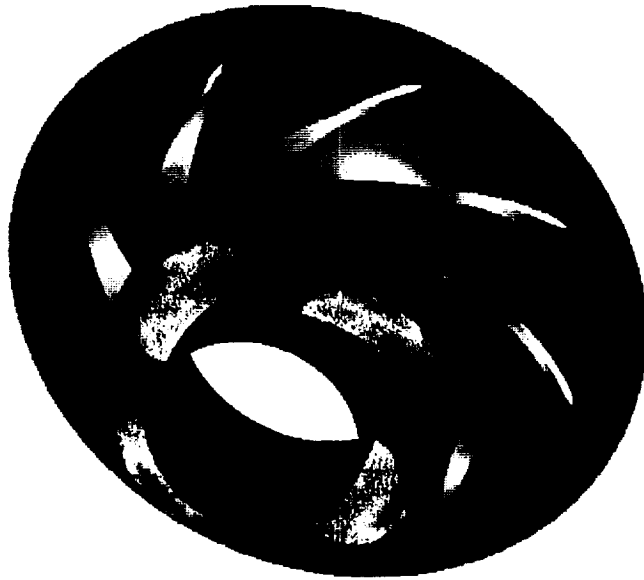
OUTLINE

- Atd inducer/impeller CFD Analysis
- Current cavitation modeling approach
- Review of cavitation approaches
- Suggestions for future modeling
- Closure

ATD 14.6 Degree Inducer-Impeller with Splitters



Suction Side Blade Pressure



Pressure Side Blade Pressure



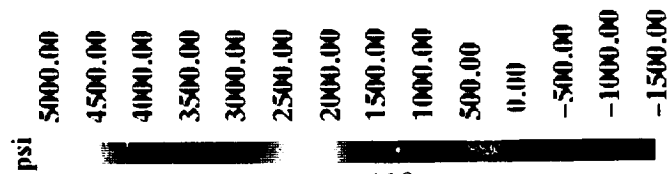
Hub Surface Pressure



Centerline Pressure



Tip Surface Pressure



CURRENT CAVITATION PROCEDURE APPLIED TO ATD CONFIGURATION

- ❑ Compute single phase 3–D CFD solution for the ATD configuration. Calculation shows pressures less than vapor pressure on suction side near the blade tip.
- ❑ Compute 2–D potential flow cavitation sheet model to obtain cavitation correction factors for *pressure loads* as a function of tip chord length.
- ❑ Scale cavitation correction factors linearly such that no correction is needed at the hub. This yields a 2–D distribution on blade surface.

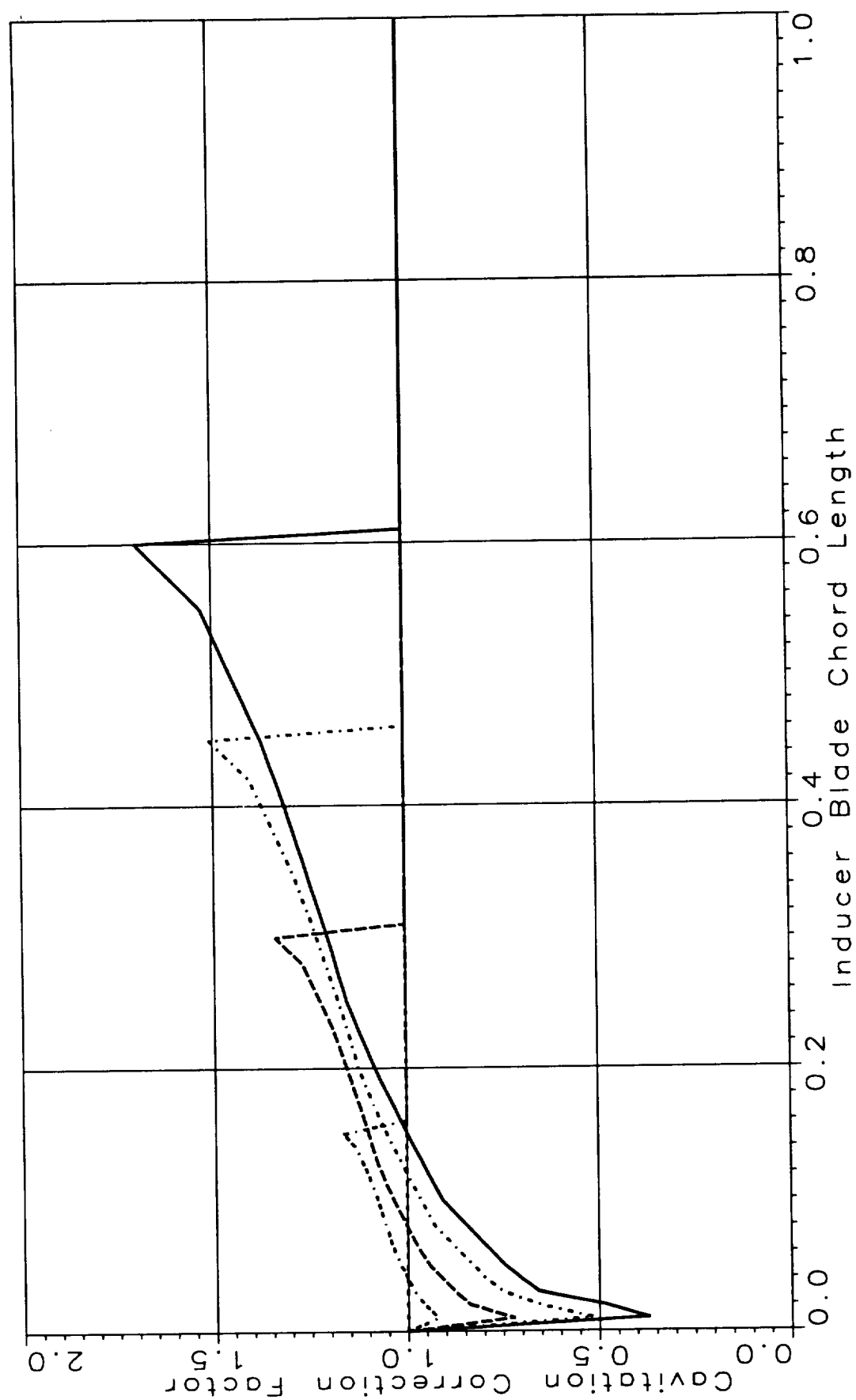
$$\Delta p_{cor} = K(s, r) \Delta p_{sheet}$$

- ❑ Transform cavitation correction factors for delta –pressure to suction side pressure in the CFD solution.

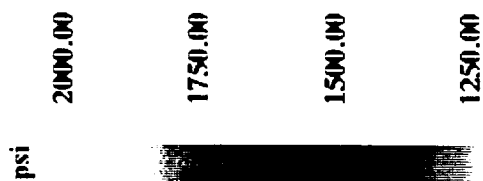
$$P_s = P_p - \Delta P_{cor}$$

- ❑ As a sanity check verify that the corrected suction pressure remains above the vapor pressure.

ATD 14.6 Inducer-Impeller CFD Predictions
Spanwise Variation of Cavitation Correction Factor



ATD 14.6 Inducer-Impeller CFD Predictions Inducer Blade Delta Pressures



No Cavitation Correction

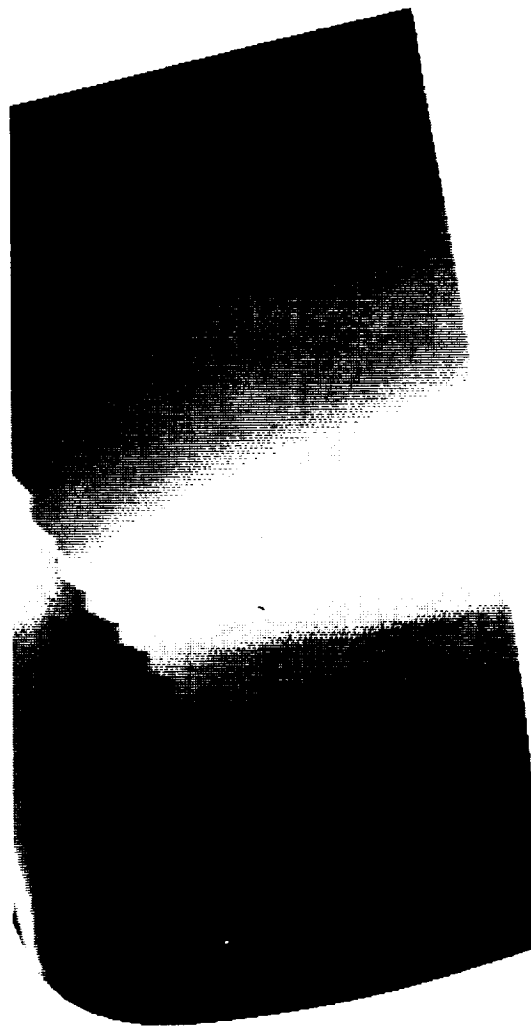


Corrected for Cavitation

ATD 14.6 Inducer-Impeller CFD Predictions Inducer Blade Suction-Side Pressure



No Cavitation Correction

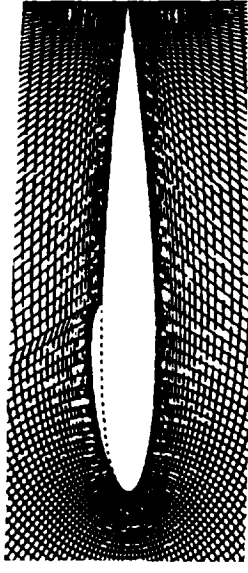


Corrected for Cavitation

LINEAR CAVITATION MODEL

- ❑ Cavity interface is treated as a streamline
- ❑ Analogous to classical linear theory used in potential flow codes
- ❑ Interface conditions are transferred to the body surface
- ❑ Zero normal velocity condition is relaxed at solid boundaries
- ❑ Approach does not treat vapor phase.
- ❑ Approach is not robust – cavity termination problems

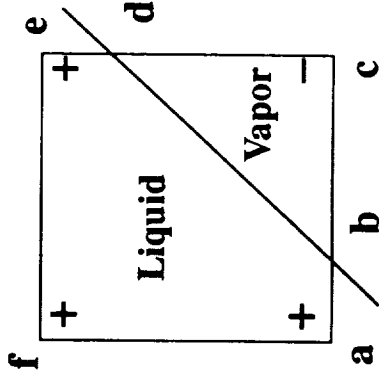
NON-LINEAR CAVITATION MODEL



- Cavity interface is treated as a streamline with a specified cavitation pressure
- Computational domain evolves with solution procedure as a sequence of linear solutions followed by modifications of the domain to accommodate cavity geometry.
- Mesh movement may introduce significant grid distortions and grid-lines may actually cross.
- Regridding may only be suitable for box-like domains and may have significant problems for complex 3-D configurations.
- 2-phase flows can be accommodated with difficulty.

PROPOSED CAVITATION MODEL

- ☐ Carry Out Flux Balance for Both Liquid and Vapor Cells Separately.
- ☐ Carry Out Flux Balance for Liquid and Vapor Parts of Interface Cells. These Cells Will Have Multiply Defined Values at Nodes.



- ☐ Nodes a, e, f Have Real Values for Liquid and Fictitious Values For Vapor.
Node c Has Real Values for Vapor and Fictitious Values for Liquid.
- ☐ Liquid and Vapor Nodes of Interface Cells Will Vary Independently.
- ☐ This Constitutes a Non-linear Multi-phase approximation to Cavitation Without the Mesh Regridding Option.

ADDITIONAL MODELING ENHANCEMENTS

- ☐ Employ a more elaborate equation of state which is approximately valid in gas/liquid two-phase region, as well as at moderately high temperature.
- ☐ Employ a more accurate energy equation to describe two-phase flow physics.
- ☐ Certain cryogenic fluids, with critical temperature below 50 K, exhibit quantum effects, and do not necessarily satisfy the principle of corresponding states.
- ☐ Embedded mesh adaptation can eliminate the need for carrying out multi-valued flux balance.

CLOSURE

- ❑ A CFD solution for ATD inducer/impeller with splitters is presented.
- ❑ Current cavitation approach employs a separate potential flow code that yields cavitation correction factors for the CFD solution.
- ❑ Penn State is studying linear and non-linear approaches for 3-D configurations.
- ❑ A suitable linear cavitation model will be implemented in the Pratt & Whitney design code.
- ❑ Additional enhancements will be considered in the future.